Hazy Octave

Building instructions

V2.0

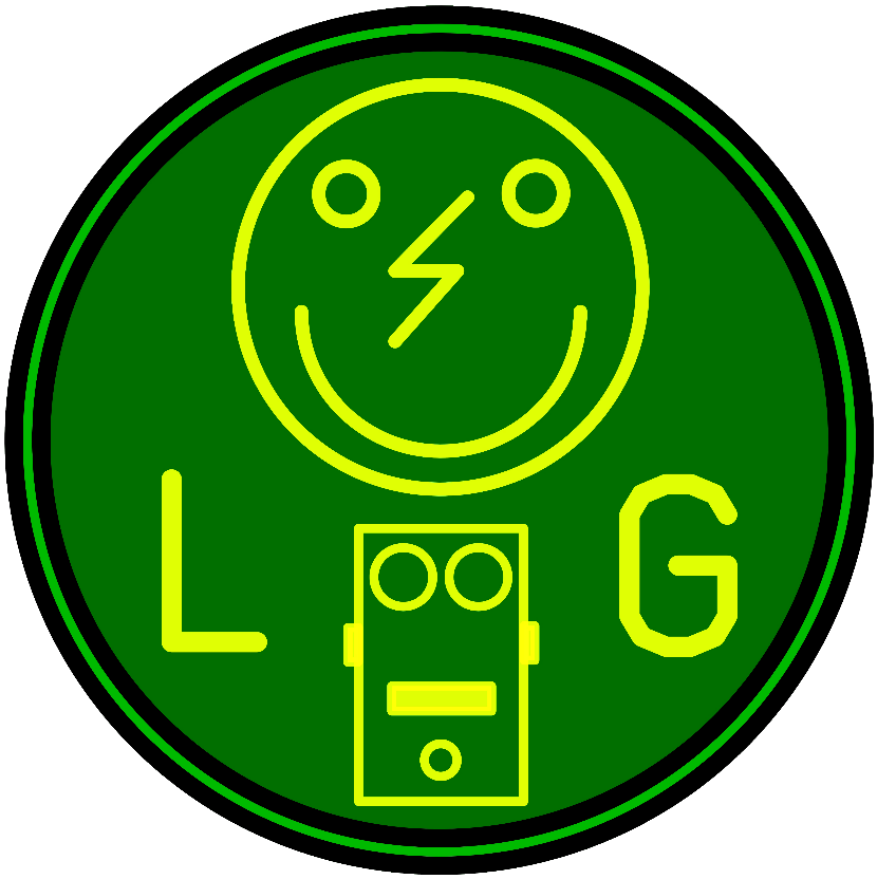


Table of contents

Changelog since v1.0

- **Pregain** rebuilt and added mod to **Smooth** control and onboard bypass switch.
- Pots are now PCB mount and pregain changed to PCB trimpot
- Diode lift switch changed to onboard switch.

Read this entire manual thoroughly before you start building the effect!

Last update: 28-03-2021

Components

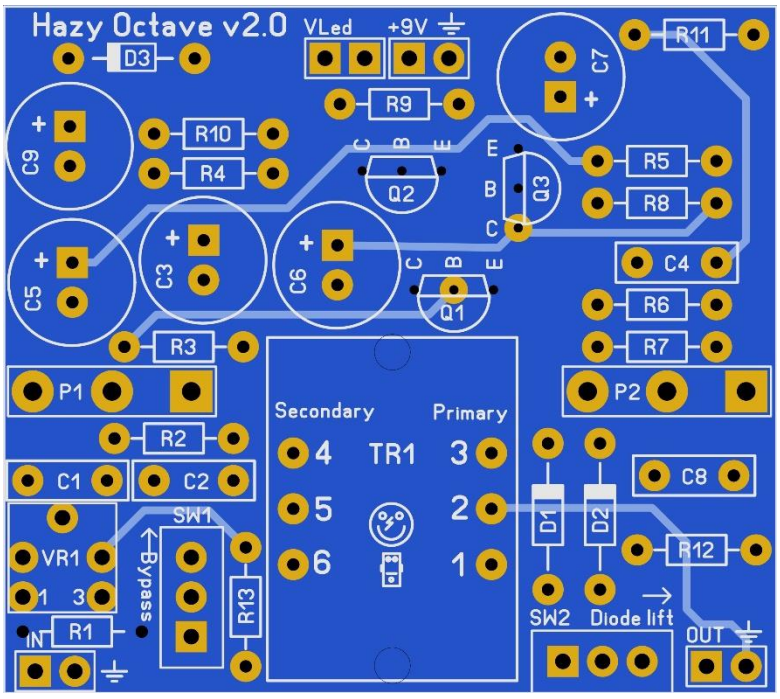
Name	Value	Comment	Name	Value	Comment
C1	100n	SMF	**R1**	2M2	(optional)
C2	150p	Ceramic disc	**R2**	680k	
C3	33u	Electrolyte	**R3**	820k	
C4	1n	Ceramic disc	**R4**	180k	
C5	22u	alternative 100u	**R5**	1k	
C6	33u	Electrolyte	**R6**	220k	
C7	220u	Electrolyte	**R7**	220R	
C8	100n	SMF (optional)	**R8**	47k	
C9	100u	Electrolyte	**R9**	22k	
D1	1N34A	Germanium Diode or 1N270	**R10**	1k2	
D2	1N34A	Germanium Diode or 1N270	**R11**	470R	
D3	1N5817		**R12**	1M5	(optional)
Q1	2N3906		**R13**	NC/0R/2k2	(optional)
Q2	2N2222A		**TR1**	42TM022	Alternative 42TU011
Q3	2N2222A		**VR1**	B50k	Pre Gain/Smooth
SW1	SPDT	Bypass pregain/Smooth	**P1**	C1k	Boost (or B1K as alternative)
SW2	SPDT/DPDT	(Foot)switch for Octave lift	**P2**	A500k	Volume
SW1/SW2 should be a PCB mounted miniature switch. **SW2** can alternatively be a DPDT/3PDT footswitch					

A=Log, B=Lin, C=Rev. Log

PCB layout

49,5 mm x 44 mm

1.95 in x 1.73 in



Bill of Materials

Capacitors		
Component	**Amount**	**Comment**
150p	1	Ceramic disc
1n	1	Ceramic disc
100n	2	SMF
22u	1	alternative 100u Electrolytic
33u	2	Electrolytic
100u	1	Electrolytic
220u	1	Electrolytic
Diodes		
Component	**Amount**	**Comment**
1N34A or 1N270	2	Germanium Diode
1N5817	1	
Transistors		
Component	**Amount**	**Comment**
2N3906	1	alternative 2N5087
2N2222A	2	alternative 2N5088, 2N4401
Resistors		
Component	**Amount**	**Comment**
220R	1	1% metalfilm
470R	1	1% metalfilm
1k	1	1% metalfilm
1k2	1	1% metalfilm
2k2	1	1% metalfilm
22k	1	1% metalfilm
47k	1	1% metalfilm
180k	1	1% metalfilm
220k	1	1% metalfilm
680k	1	1% metalfilm
820k	1	1% metalfilm
1M5	1	1% metalfilm
2M2	1	1% metalfilm
B50k	1	Bourns 3362P or similar
C1k	1	Reverse Logarithmic 1k PCB potentiometer
A500k	1	Logarithmic 500k PCB potentiometer
Transformer		
Component	**Amount**	**Comment**
42TM022	1	alternatives 42TU011 or 42TM011

Introduction

The Hazy Fuzz is based on the Tycobrahe Octavia™ (which itself is based on the Roger Mayer Octavia™). It has been specially adapted to fit on a small PCB and include all sorts of optional modifications. Mods include a pre-gain/smooth and octave lift switch as well as true bypass with pull down resistors. Read the modifications section on how to get as close as possible to the original.

Building sequence

Soldering this board can be very complicated for some people since the solder pads are very close together. Use a magnifying glass to make the job easier. If you want to experiment with other transistors then you could socket them instead of soldering them to the board. You'll need a 20 SIL, break off the sockets and solder them to the board.

Start by soldering the jumpers (if needed), resistors and diodes **D1** and **D2**. Warning: Germanium diodes are fragile! Do not heat them to long or they will break.

If you want to socket the transistors then solder the sockets now. **Note:** Do not blow on your solder in an attempt to cool it down. That will possibly result in a bad join that might corrode!

Solder the ceramic capacitors next and then the small SMF capacitators, then the electrolytes, finish by soldering the transformer to the board.

Place the transistors and you are almost ready to rock.

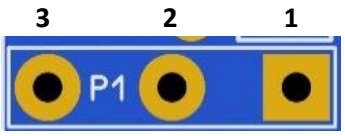
Besides the components mentioned in the Bill of Materials table, you will need:

- **2 input jacks**. 2 mono jacks if you are not going to use a battery but only the 9V adapter. 1 mono (for output) and 1 stereo jack (for input) if you will be using both a 9V battery and the 9V adapter.
- **3PDT footswitch** (9 pins).
- **2,1mm DC jack** (isolated).
- **9v battery clip** (optional).
- **22 gage stranded hook-up wire.**
- **2 x LED.** Only 1 if you are not implementing the octave as a footswitch (**SW2**)
- **Hammond 125B** case (or similar) in your favorite color.

Off board wiring

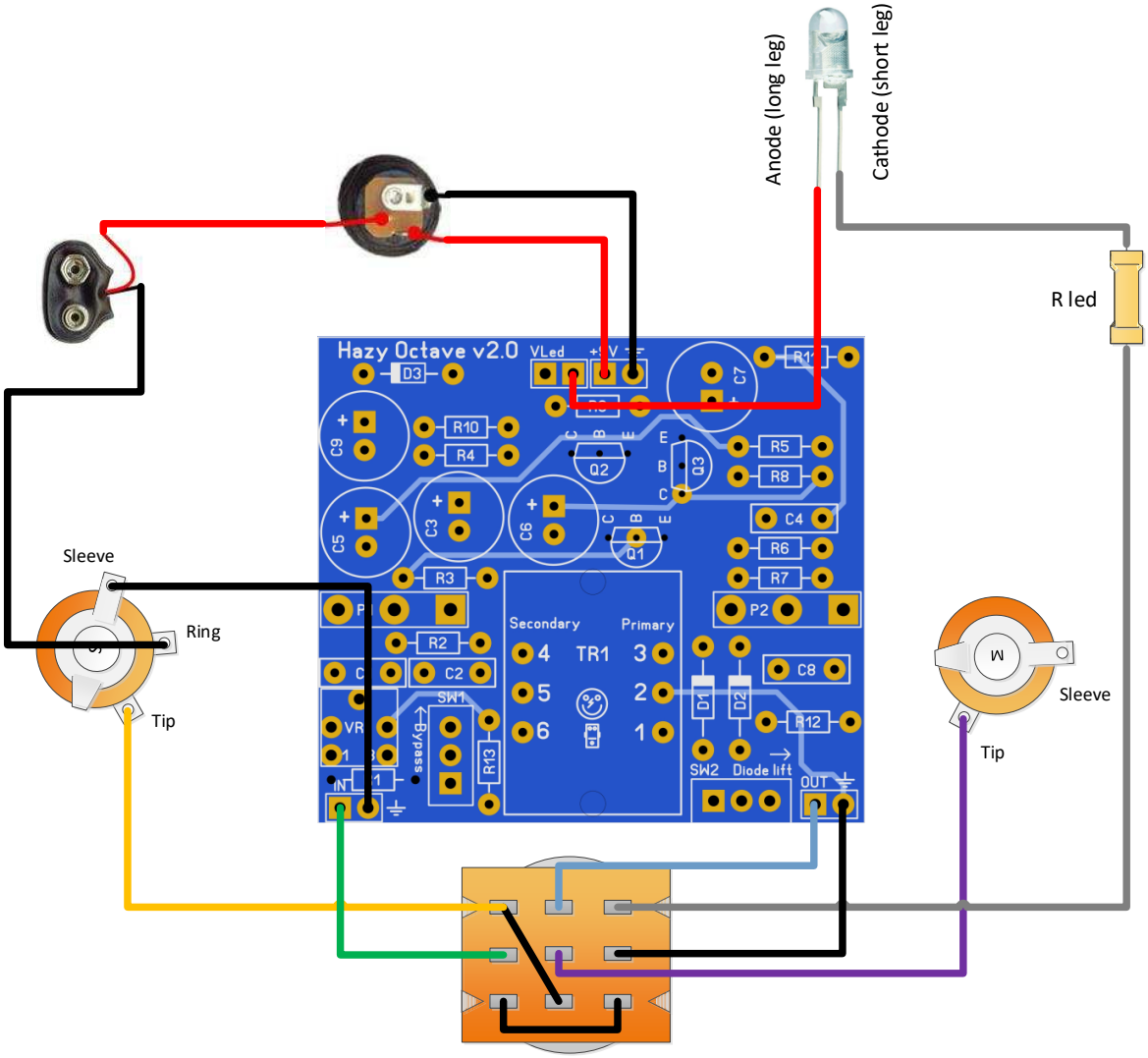
Wiring the pots **P1**-**P2** is very simple as they are PCB mounted pots to the backside of the PCB. The rectangle pad marks pin 1 of a potentiometer. The images below show how you can recognize which pin is which on a potentiometer.

Blue = pin 1
White = pin 2
Yellow = pin 3



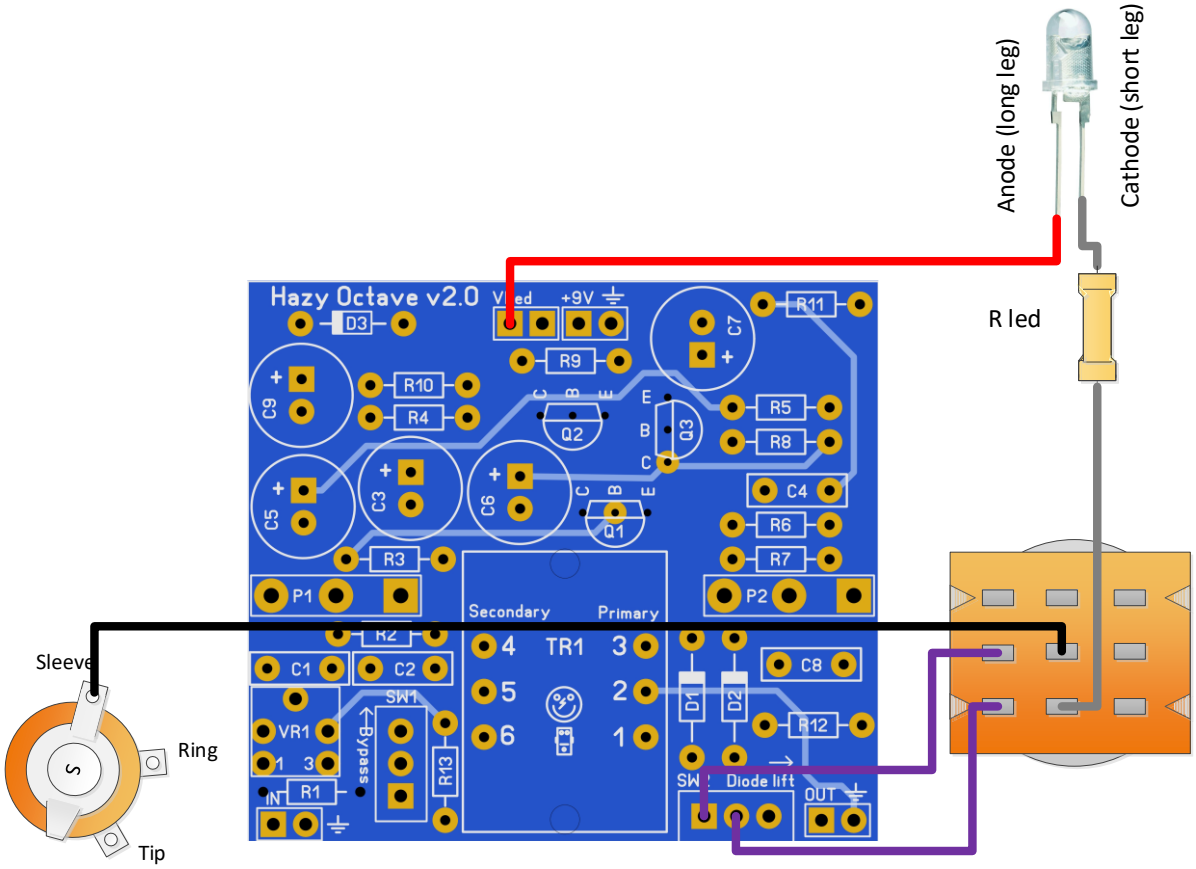
Now you can wire the rest of the board, keep the wires as short as possible to prevent noise.

Basic wiring



Diode lift footswitch

If you prefer to use the diode lift switch (**SW2**) as an external switch then wire it like this:



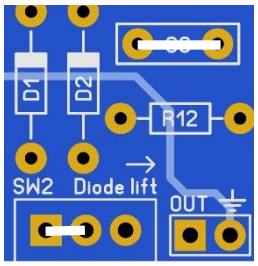
Also note that if you do not want to use the diode lift switch (**SW2**) then jumper the middle pad of **SW2** to the rectangle pad with a spare piece of wire (eg from a resistor).

R led is the current limiting resistor to the LED. The value of this resistor varies depending on the type of LED you are using. If you are using a clear ultrabright LED you can best use a 4k7 to be safe, but feel free to experiment. The lower the value the brighter the Led will glow and the shorter its lifespan.

Modifications

Stay original

The layout of the board is made to incorporate as much modifications as possible. If you want to get as close as possible to the original than leave out **VR1** (pre-gain), **SW1** and jumper pads 2 and 3 on **SW1**. Also do not install **R1** and **R12** (pulldown resistors), **R13**, **C8** (DC filter), **C9**, **D3** and the octave switch **SW2.** You need to short pad 1 and pad 2 of **SW2** and jumper **C8** and **D3**.



Diodes

D1 and **D2** need to be Germanium diodes. Preferably 1N34A or alternatively 1N270. It's been mentioned that you should try to match the forwarding voltages (U_f) of both diodes for an optimal result.

Boost mod

The Boost is reported to perform better with a 100uF instead of the default 22uF in **C5**. If you want to experiment with different values as **C5** then solder 2 SIL pins in **C5**.

Transistors

You can also use different mixes of transistors. I encourage you to experiment with this because it is always a taste question. Some mixed will differ more profound than others. Mixes you could try for **Q1**, **Q2** and **Q3**:

- 2N3906 and 2N2222A (2x)
- 2N5087 and 2N4401 (2x)
- 2N5087 and 2N5088 (2x)

Pregain vs Smooth

To be honest there is not a great deal of difference in tone for both setups. It is again a question of personal preferences.

Pregain

R13 need to be shorted by a jumper or if you want it to have a minimum value, use a 2k2 for **R13** (or another value to your liking)

Smooth

Simply leave out **R13**. No need to jumper it!

Troubleshooting

All PCB's have been 100% factory e-tested and out of every batch I receive I build an effect to double check, so there should not be a connection problem on the PCB itself.

The board is not working (at all), what now?

- Check if your 9V is plugged in correctly (and/or soldered correctly on the board).
- Check that you oriented the capacitors, IC's ,transistors and diodes the right way. SMF, MKT and ceramic capacitors as well as resistors do not need to be oriented.
- Check if you used the correct values of the components. For resistors you can look here: http://www.diyaudioandvideo.com/Electronics/Color/
- Double and triple check your soldering! A loose or cold solder can be really bad for your board.
- Replace the IC, it might be defective. Before doing that first unplug the 9V and wait for 5 seconds.
- Check that you have good/high grade components. A lot of Chinese sourced parts are fakes (especially high end opamps, vintage diodes and transistors) so be careful that you source your parts from reliable suppliers.

Schematic